SYSTEMATIC APPROACH TO ANALYZING AND REDUCING AERODYNAMIC DRAG OF HEAVY VEHICLES

Rose McCallen (925) 423-0958

Dan Flowers (925) 422-0529

Lawrence Livermore National Laboratory, Livermore, CA

Walt Rutledge (505) 844-6548

Kambiz Salari (505) 844-9836

Sandia National Laboratories, Albuquerque, NM

Anthony Leonard (626) 395-4465

California Institute of Technology, Pasadena, CA

Bob Englar (770) 528-3222

Georgia Tech Research Institue, Atlanta, GA

Fred Browand (213) 740-5359

Mustapha Hammache (213) 740-5377

University of Southern California, Los Angeles, CA

Jim Ross (650) 604-6722

Bruce Storms (650) 604-1356

NASA Ames Research Center, Moffett Field, CA

Summary

This poster describes DOE's multi-laboratory, multi-university effort to reduce aerodynamic drag of heavy vehicles by developing and demonstrating new approaches for simulation and analysis of aerodynamic flow around heavy truck vehicles. Greater use of newly-developed computational tools holds the possibility for reducing the number of prototype tests, cutting manufacturing costs, and reducing overall time to market.

Experimental validation are also an important part of this approach. Experiments on a model of an integrated tractor-trailer are underway at NASA Ames Research Center and the University of Southern California. Companion computer simulations are being performed by Sandia National Laboratories, Lawrence Livermore National Laboratory, and California Institute of Technology using state-of-the-art techniques. In addition, aerodynamic devices are being investigated by Georgia Tech Research Institute.

Work performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract W-7405-ENG-48 and Sandia National Laboratories, University of Southern California, and California Institute of Technology.

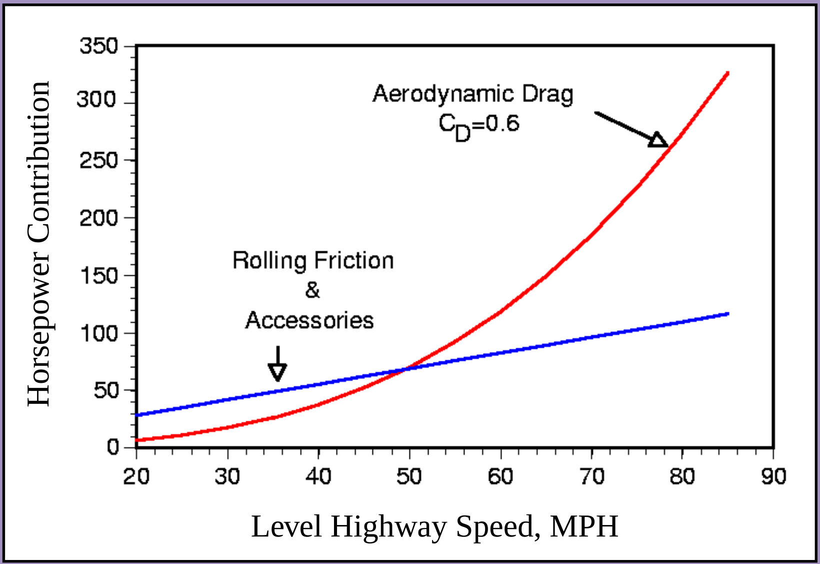
Motivation

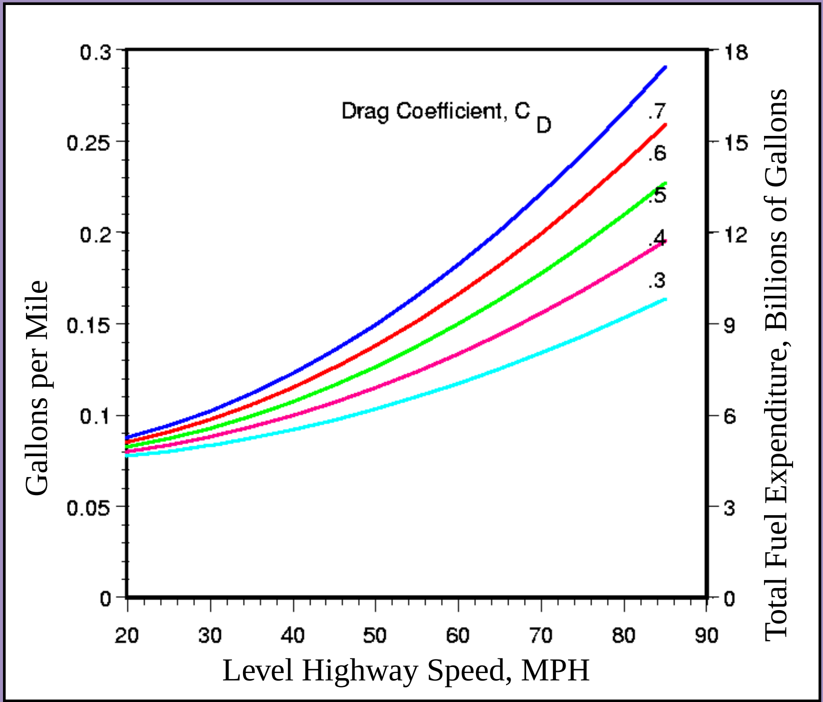
Reducing Fuel Consumption by Reducing Aerodynamic Drag

The achievement of reduced fuel consumption hinges upon the availability of trucks having greater aerodynamic efficiency. In the past twenty years, drag coefficients for typical large trucks have decreased by about 30% -from the range C_D=0.8-1.0 to C_D=0.5-0.7. Economical travel will require even greater efforts to integrate tractor and trailer design in one clean aerodynamic-package.

Reducing C_D from 0.6 to 0.3 would result in a total yearly savings of 3 billion gallons of diesel for travel at 60 mph - a 43% savings.

Overcoming drag represents 65% of the total energy expenditure at a speed of 70 mph.





Class 8 tractor-trailer
weight = 80,000 lbs

$$C_D = \frac{\text{drag force}}{\text{dynamic pressure} \cdot \text{projected area}}$$

Goals and Objectives

DOE Workshop on Heavy Vehicle Aerodynamic Drag
Phoenix, Arizona
January 30-31, 1997

Conclusions

Trailer design should be the focus of near term efforts.

An integrated tractor-trailer design is needed to achieve significant drag reductions.

Advanced computational tools are needed to efficiently determine appropriate design changes.

Project Goal: **Develop and demonstrate the ability to analyze aerodynamic flow around heavy vehicles using *existing* and *advanced* computational tools.**

Project Objectives: **Provide computational tools for effectively and efficiently designing and analyzing vehicle components and future fully integrated tractor-trailers.**

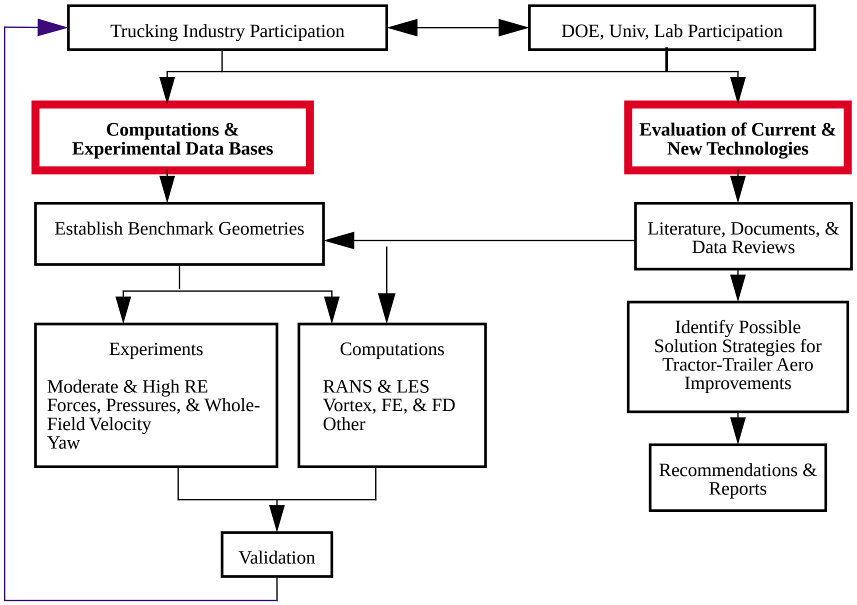
At present, the aerodynamic design of heavy trucks is based largely upon wind tunnel estimation of forces and moments, and upon qualitative streamline visualization of flow fields. Computational analysis tools can reduce the number of prototype tests, cut manufacturing costs, and reduce overall time to market.

Approach

The project is divided into two related and overlapping efforts:

> **Advanced computations and experiments of benchmark geometries**
>
> **Evaluation of current and new technologies**

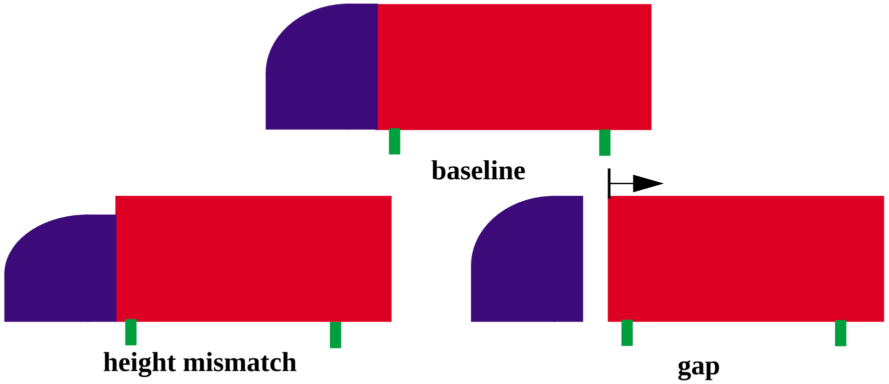
Each effort has near-term deliverables as well as longer-term goals. The computations and experiments will provide rapid results for simple benchmark geometries, and will advance to more complex geometries. The evaluation of current and new technologies will provide continued assessment for promising emerging technology.



FY99 - FY00 **Simulation and experimentation of a simple integrated tractor-trailer model**

Investigate the effects of tractor-trailer height mismatch and gap distance

Visits with industry, workshop, documentation and publication

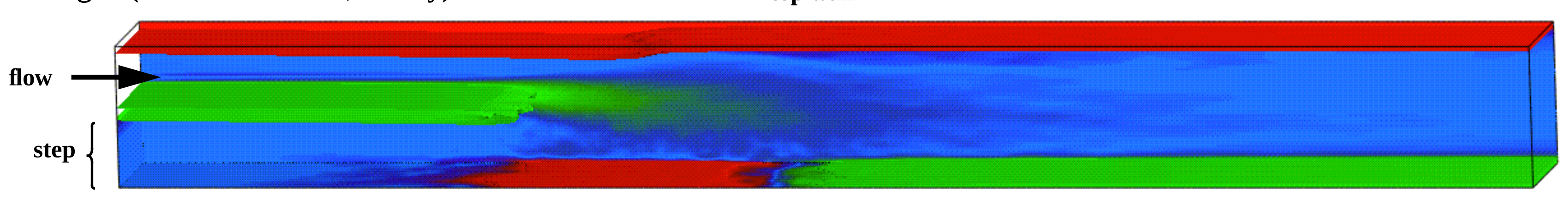


Computational Tools

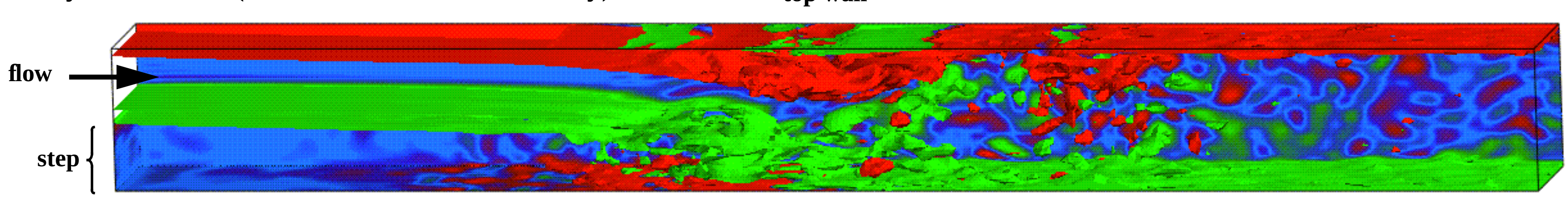
Existing Computational Tools	Provide **time-averaged** flow predictions
(Reynolds-Averaged Navier-Stokes)	Utilize models that have empirically based parameters
	With proper experimental validation, these tools can provide some design guidance
	Currently available
Advanced Computational Tools	Provide full three-dimensional simulations of time-dependent flow
(e.g., Large-Eddy Simulation)	Less empiricism
	Computationally intensive
	In development

Comparison of time-averaged and large-eddy simulations of a wall bounded flow with a backward-facing step at the entrance.

Time-Averaged (two-dimensional, steady)



Large-Eddy Simulation (three-dimensional, unsteady)

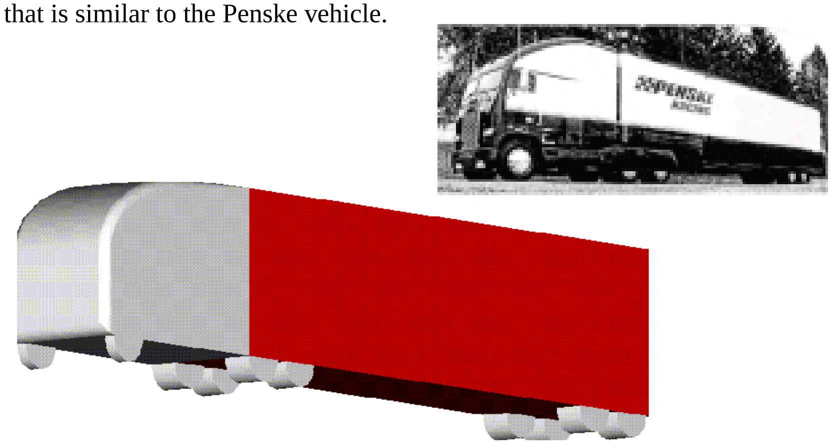


Computational Prediction of Aerodynamic Drag for a Simplified Truck Geometry

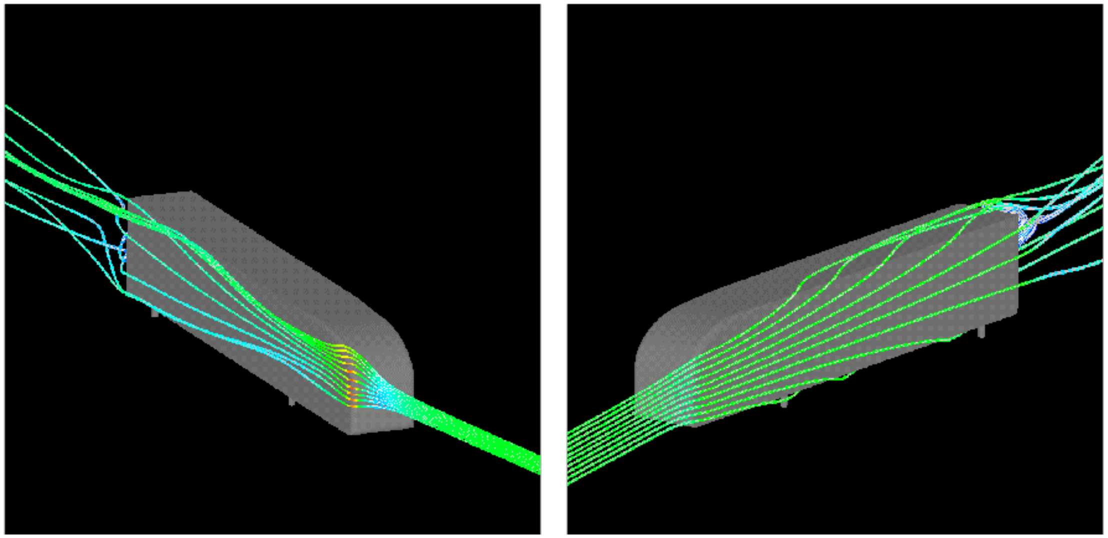
Reynolds-averaged Navier Stokes (RANS) computations are being performed by Sandia National Laboratories (SNL). The figure below shows a flow field simulation about the Ground Transportation System (GTS) Model at Reynolds number of 1.6 million at 10 degree yaw angle. The presented results involve the modeling of an experiment performed on the GTS Model in the Texas A&M 7-ft. x 10-ft. wind tunnel. These RANS simulations include part of the converging section, test section, and part of the expansion region of the tunnel. The tunnel walls are treated as slip boundaries (no penetration). The computational meshes for the RANS simulations range from a coarse mesh of 0.5 million nodes, a medium mesh of 4 million nodes, and a fine mesh of 32 million nodes at 0 and 10 degree yaws. For these calculations an implicit finite-volume compressible flow solver with a one-equation Spalart-Allmaras turbulence model was used. The steady solutions were obtained on a massively parallel machine using 107 and 246 processors for the coarse and medium mesh, respectively. The fine mesh calculation which uses 1414 processors is underway. Future plans are to use these solutions as the initial conditions for time-accurate RANS calculations.

Sandia National Laboratories

Solid model of GTS tractor-trailer baseline geometry that is similar to the Penske vehicle.

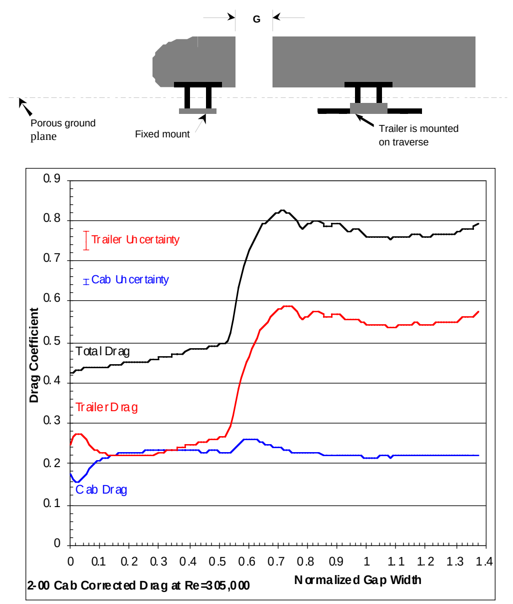


The 10° yaw, medium mesh, particle traces are colored by velocity magnitude.



Tractor-Trailer Gap: The Relationship Between Measured Drag and Measured Flow Field

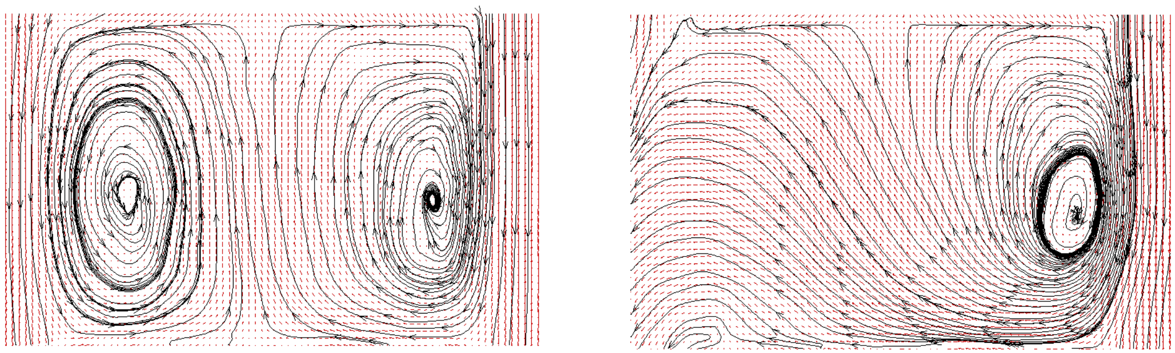
Wind tunnel measurements of drag are conducted using 1/14 scale models fabricated from styrofoam using a rapid prototyping 3-axis milling machine. The plot below illustrates the dramatic effect of gap length upon drag. Minimum drag occurs for zero gap, and there is a gradual increase in drag as gap increases in the range G/L=0-0.5 (G is the gap width and L is the square root of the frontal area). At G/L= 0.5-0.6, there is a sudden increase in the drag of the trailer. The flow also becomes much more unsteady in the vicinity of this critical gap, suggesting that a different flow regime is somehow established. The details of the flow field within the gap are studied using planar Digital Particle Image Velocimetry. The technique captures the instantaneous image of many particles illuminated by a laser light sheet. Approximately 25 microseconds later, a second laser is fired, and a second image acquired. The two captured images are interrogated locally to determine a single displacement field (velocity field) consisting of approximately 5000 vectors. Approximately 50 such realizations are acquired for each gap length, and for each vertical or horizontal slice. The results show that at short gap lengths (G/L< 0.4) , the individual realizations describe a relatively steady flow containing a stable toroidal vortex within the gap. For large gap lengths (values of G/L > 1.0), the gap can no longer support the steady vortex, and vorticity is continually shed downstream. Near G/L~0.5, the flow alternates between these two states. This alternation is illustrated in the streamline plots below, showing two horizontal slices at mid height. The two "states" are separately detected and averaged. The image on the left illustrates the nearly symmetric flow that is present part of the time, while the image on the right represents a portion of time having strongly asymmetric right-to-left flow within the gap.



Horizontal flow slices
G/L = 65%

Symmetric

Asymmetric

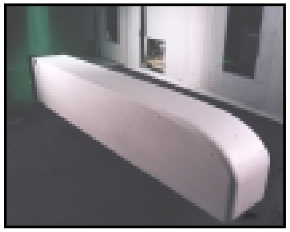


3D Particle Image Velocimetry (3DPIV) of a Truck Wake

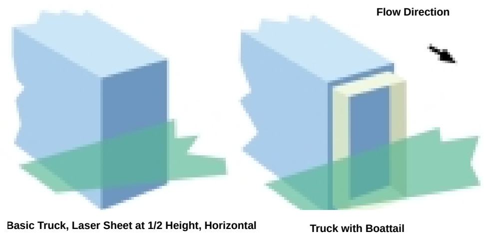
Particle Image Velocimetry is an imaging technique that measures both the velocity and direction of fluid flow on a given plane in space. 3DPIV data has all three components of a velocity vector. This technique uses a pulsed dual-head Nd:VAG laser whose output is formed into a sheet of light. This beam which is less than a millimeter thick, is projected into the fluid flow of interest. The laser illuminates seed particles that get introduced into the fluid. In the case of this experiment, the seed is atomized mineral oil. For an accurate measurement, the seed must follow the flow without “lagging” behind. The laser can make two successful light pulses with the delay between pulses tightly controlled. Two cameras view the laser sheet such that they form a stereo-pair. They record the particle field as illuminated by the two pulses. Image processing of the reference and delay images measures the shift in the parrticles in the time between the pulses. Each camera will yield a two-component vector field. Further data processing yields a single vector field, derived from the 2D fields, that has the third directonal component.

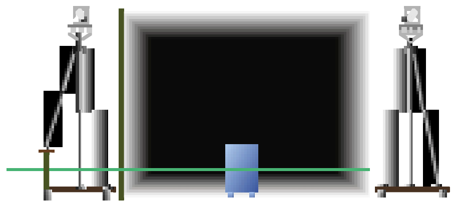
National Aeronautics & Spaceation

Installation view of truck in Ames 7x10 wind tunnel

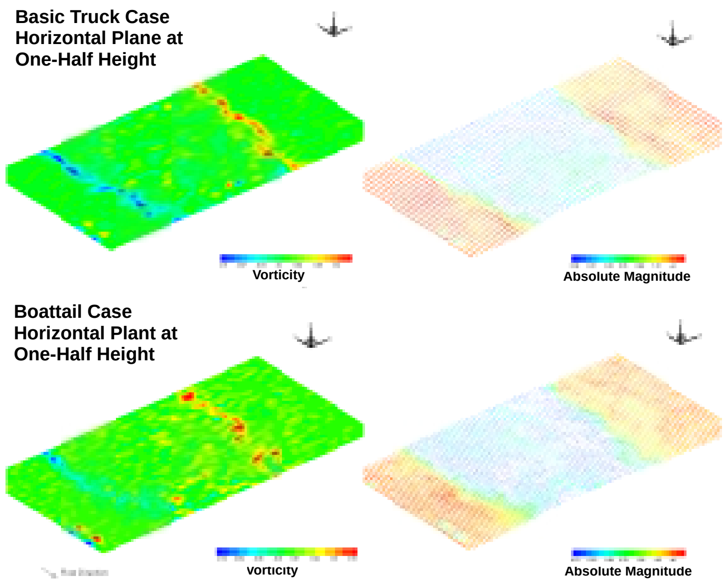


Points of view for data



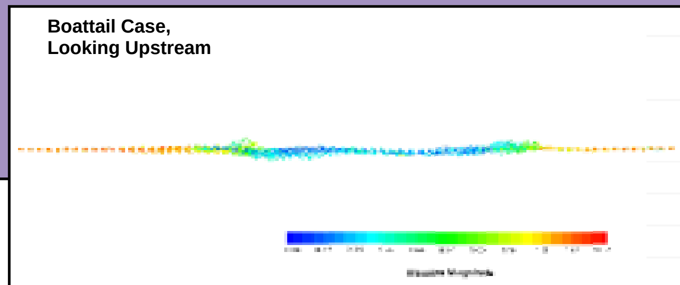


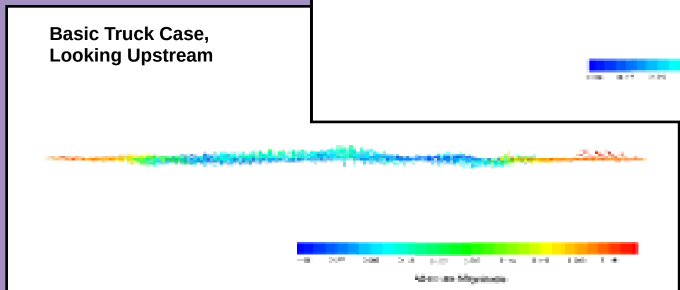
Cameras and model (looking upstream)



Vector and vorticity data of the two truck configurations
Plots show the difference in both the wake contraction and turbulence level

Views showing the third component





A Computational Study of the Influence of Boattail Plates on the Trailer Flow Field

University of California
Lawrence Livermore National Laboratory

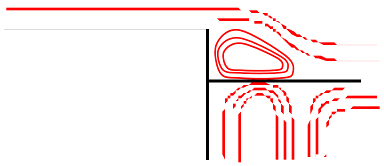
Aerodynamic drag can be significantly reduced with trailer add-ons that reduce the wake and increase the base pressure. Continuum Dynamics, Inc. has provided boattail plates to fit the GTS model for tests in the NASA Ames 7-ft. x 10-ft. wind tunnel. Tests conducted with and without the boattail plates show a 20% reduction in drag when the plates are installed.

The large-eddy simulation (LES) approach is being used by Lawrence Livermore National Laboratory (LLNL) to simulate the back-end of the trailer with and without the boattail plates, as well as to simulate the entire GTS flow field. This advanced modeling approach has the potential to achieve more accurate simulations with minimum empiricism and thus, reduce experimentation. The flow around a tractor/trailer is time dependent, three-dimensional with a wide range of scales (i.e., the largest scale is on the order of the truck length and the small scales are smaller than the diameter of a grab handle). LLNL is utilizing an established finite element method in conjunction with LES to accurately capture the complex flow around the GTS vehicle.

Full-scale truck in wind tunnel

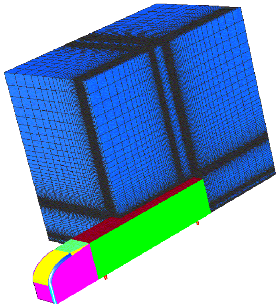
Model in wind tunnel

Plates developed by Continuum Dynamics, Inc.

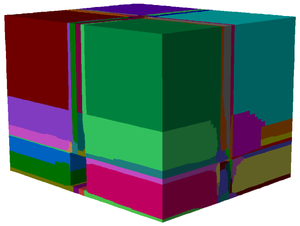


Boattail plates are designed to reduce wake size.

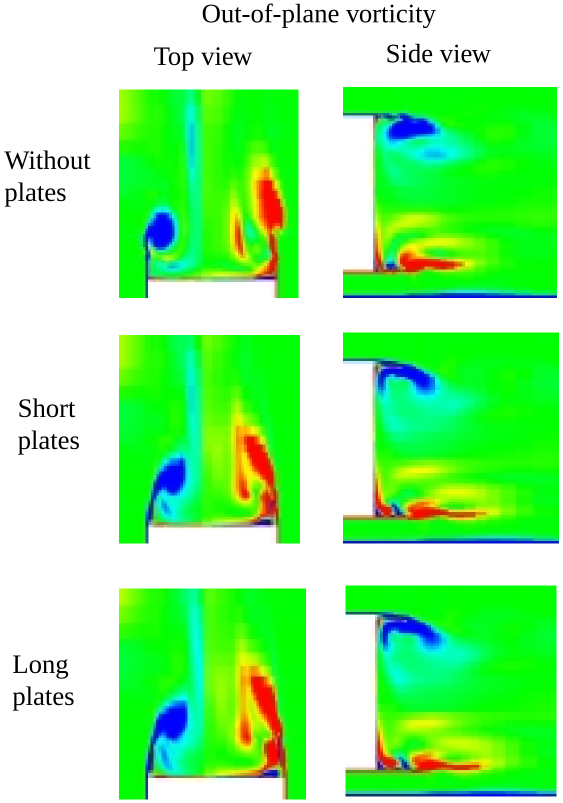
Computational Tools Can be Used to Investigate the Effect of Lengthening the Plates



Compressible flow simulation
Half of 3 million element grid

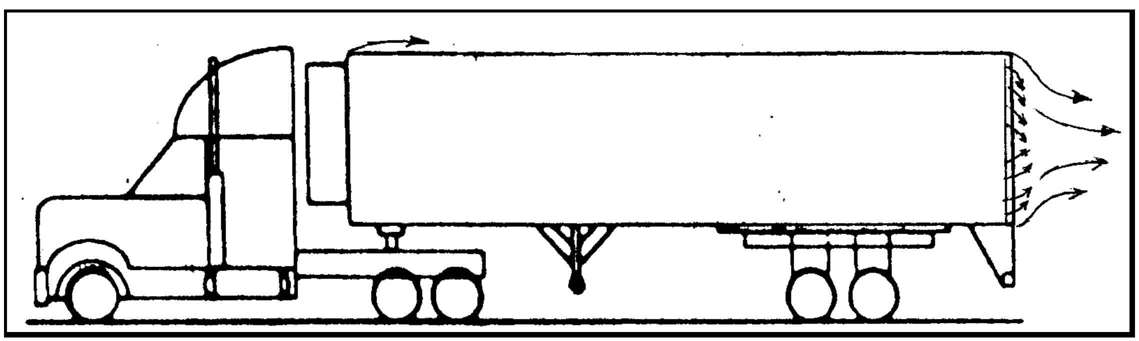


Domain decomposition
- 148 computational domains
- 148 processors on ASCI Blue massively parallel machine (IBM)



Development of Pneumatic Aerodynamic Devices to Improve the Performance, Economics, and Safety of Heavy Vehicles

The Georgia Tech Research Institute (GTRI) is developing and evaluating pneumatic aerodynamic devices to improve the performance, economics, stability and safety of operation of heavy vehicles. The objective of this research effort is to apply the pneumatic aerodynamic aircraft technology (previously flight tested, developed and patented by GTRI) to the design of an appropriate blown tractor-trailer configuration. This program includes use of appropriate analytical codes for pneumatic configuration design; experimental wind-tunnel evaluation of this blown concept and a representative baseline tractor-trailer; analysis of the resulting data; and transfer of that technology to the development of a full-scale road-test demonstrator. Anticipated overall results to be confirmed will be the pneumatic augmentation or reduction of aerodynamic forces and moments as desired by the vehicle's driver or automated control system, and the resulting increased performance, economics, stability, and safety of that vehicle's operation. Geometry and blowing system requirements for these devices are being determined, and a preliminary systems study is being conducted. The pneumatic model will include devices to: reduce aerodynamic drag for efficiency or increase drag for braking; increase lift (and thus reduce tire rolling resistance) or reduce lift (and thus increase traction and braking); and provide increased lateral/directional stability and safety, all without use of external moving aerodynamic components.



Application of pneumatic aerodynamic technology to heavy vehicles currently being conducted at GTRI

Flight tests confirming pneumatic devices for aerodynamic force augmentation or reduction

Experimental confirmation of pneumatic technology on streamlined car at GTRI, showing blown jet turning on bluff aft contour

Simulation of Complex, Unsteady Flows Using a Grid-Free Vortex Method

Caltech
California Institute of Technology

The objectives of the research are to develop computational techniques, capable of simulating complex, three-dimensional, unsteady wakes at high Reynolds numbers (Re) and to apply these techniques to flows relevant to tractor-trailer aerodynamics. In addition, and in collaboration with experimental efforts in the overarching DOE program, a systematic study of various unsteady tractor-trailer wake flows will be investigated to increase our understanding of the complex interplay between body geometry, wake vortex dynamics, and body forces and moments. It is anticipated that such techniques will be useful as design aids to the tractor-trailer manufacturing industry.

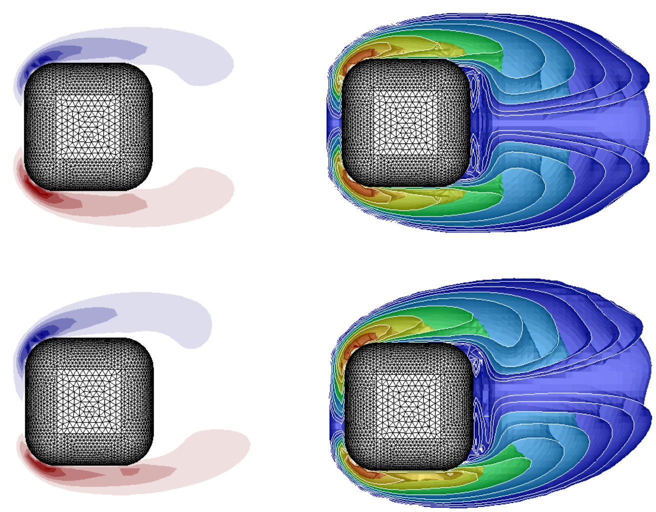
A Lagrangian vortex method, coupled with an appropriate vortex panel method, is used as a basis for the numerical scheme. In medium to high Reynolds number applications, the vortex method will function as a large-eddy simulation (LES) technique. Thus, subgrid-scale models will be required (1) to represent the effects of fine-scale turbulence not resolved by the vortex particles, (2) to represent the effects of small-scale active/passive flow control devices that may be applied, and (3) to represent small-scale perturbations to the surface of the body.

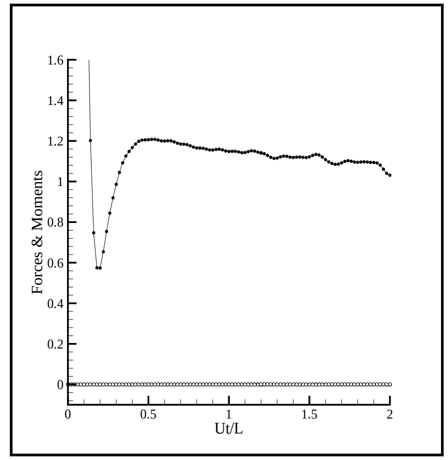
Starting flow around a rounded cube at Re=100 and Ut/L=2. Results are for a slice at the body half-height.

Out-of-plane vorticity | Vorticity magnitude

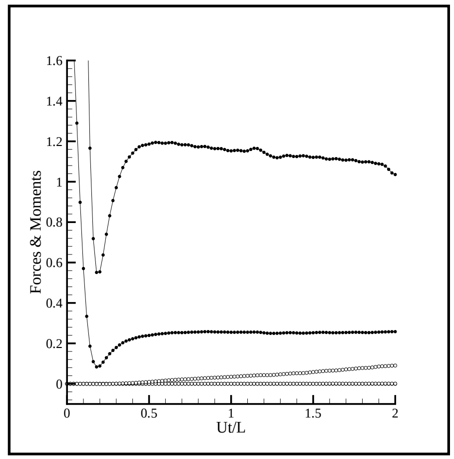
0° yaw

10° yaw





Body axis drag coefficient versus time for 0 degrees yaw. The only body force present is the body axis drag; all other forces and moments are zero.



Body forces and moment versus time for 10 degrees yaw. From top to bottom: body axis drag coefficient, body axis side force, and moment about the transverse body axis. All other forces and moments are zero.